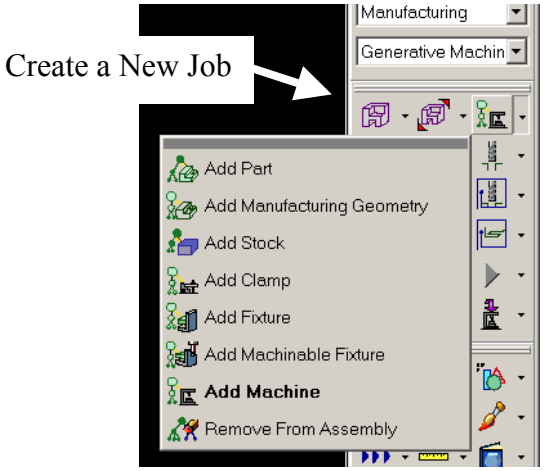
I-DEAS Generative Machining Notes
Master Series 9

by R. Link

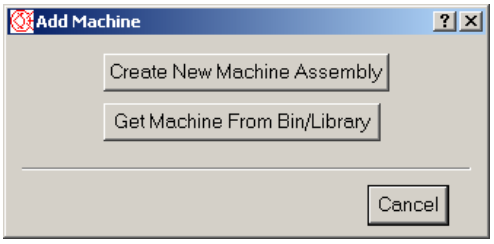
A manufacturing job in I-DEAS consists of one or more Setups. A Setup is an assembly of the part, the stock, the clamps and fixtures and the machine tool. Within the Setup are groups of Operations called Opgroups. An Operation is one specific machining process that is performed on the stock with a single tool and set of machining parameters. The Generative Machining Task is organized to let you specify the assembly and the sequence of operations necessary to produce the features that make up the finished part in a logical sequence.

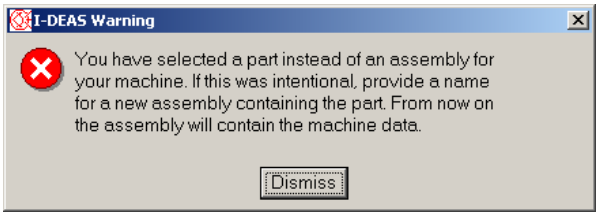
To start a manufacturing job, put all of your parts and assemblies away in your bin. Save your model file and switch to the Manufacturing Application and select the Generative Machining Task. Pick the top left icon, Create a New Job, and enter a job name at prompt or accept the default name. A message appears in the List region that says "Note: A job was created along with a default setup and opgroup."

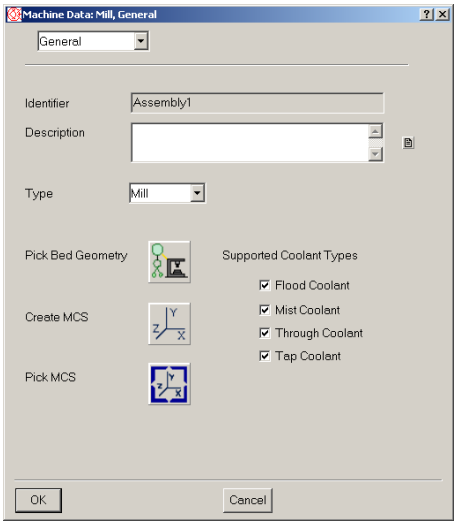


Using the stack of icons in the first row, third column, add the part to be machined, starting stock, any clamps and fixtures, and a machine. Pick the icon, click the RMB, then select, Get… From Bin and pick the item from the list. Don't worry about positioning yet.

When you select the *Add Machine* icon, select *Get Assembly from Bin/Library*. Pick the machine tool from the Bin. A warning box will appear and you can dismiss it.

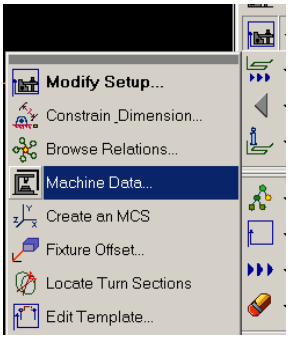






The *Machine Data* form appears. Select the type of machine tool, either Mill or Turn. At this point, the instances in your manufacturing assembly are probably not positioned correctly. Just accept the defaults by picking OK. You will be able to define a logical machining origin after the assembly is properly oriented.

After all of the instances have been added to the Setup-Assembly on the workbench, use the Move, Rotate and Align icons to position all of your assembly instances in proper position.

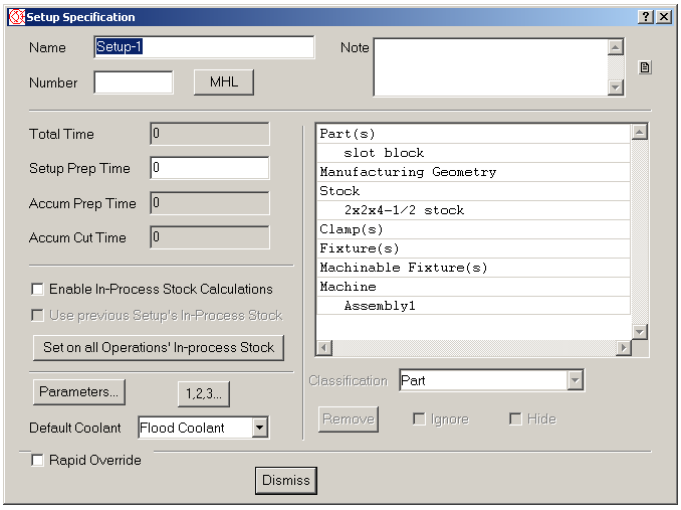


Once everything is properly positioned, create a coordinate system to serve as the origin for the machining operations.

Pick the Machine Data icon. On the Machine Data form, Select *Create MCS*. On the pop-up menu, pick Origin, next pick a convenient point on your Part to use as the origin. In general, pick an easily located point on top surface if possible. (Good candidates for locating the origin are the corner of the part or the center of a drilled hole. For turning jobs, always select the origin as the point on the centerline of the part farthest away from the chuck.)

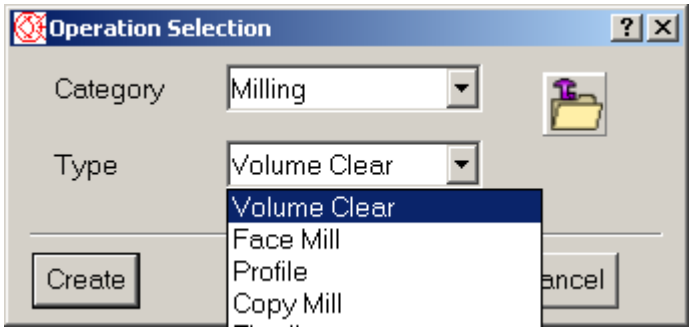
Continue using the options from the pop-up menu to set the x,y, and z axes to correspond to convention for milling or turning.

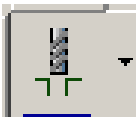
The next step is to identify this new coordinate system (CS) as the origin for the machining operations. Click on the *Select MCS* icon and pick the origin you just created. Click the OK button to dismiss the form.

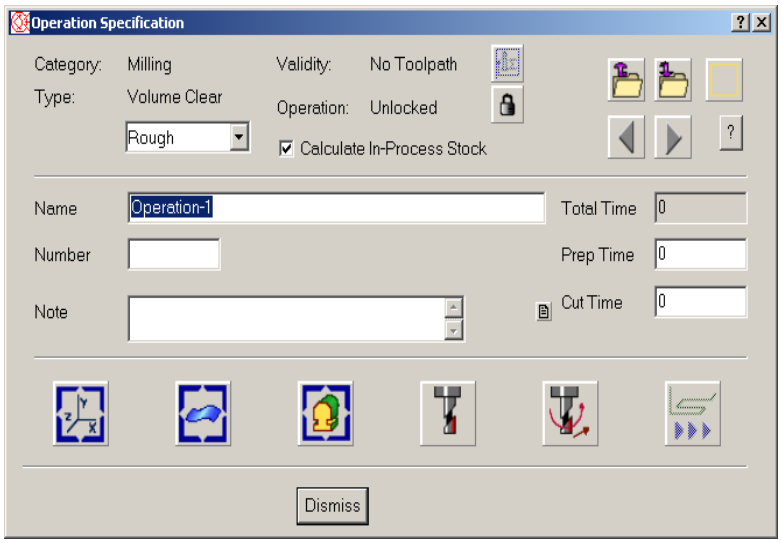


Pick *Modify Setup Specification* icon (3rd row, 1st col.) to display the Setup Specification form. On this form, check *Enable In-process Stock Calculations* box and click on *Set on all Operations In-process Stock* button, then dismiss.

The Setup has been completely defined and it is now time to define the sequence of operations needed to produce each of the features on your part. Usually you will have a series of roughing operations followed by finishing operations. You may also have some hole-making operations as well.



To add an operation, pick  and pick the appropriate category and type of operation from the drop-down menu under the Type. Then pick the Create button.



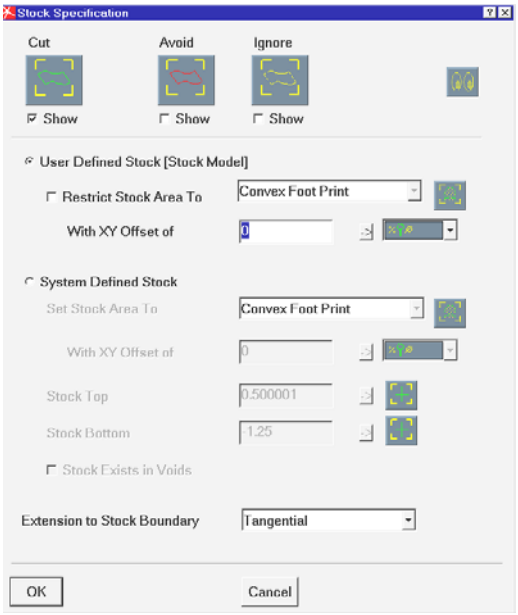
A new form like that shown here will display that allows you to completely specify all of the details defining the operation - surfaces to cut, tool to use, cutting parameters and finally, calculating the toolpath.

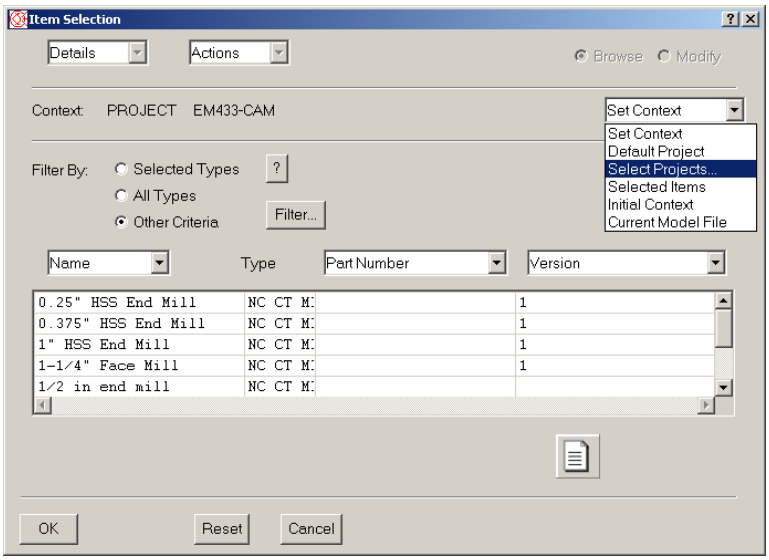
Start out by specifying a rough or finish operation.

Make sure the *Calculate In-process Stock* box is checked.

You can assign a meaningful name to the operation. The 2nd, 4th, 5th and 6th icons will usually be picked in sequence to define the rest of the operation parameters.

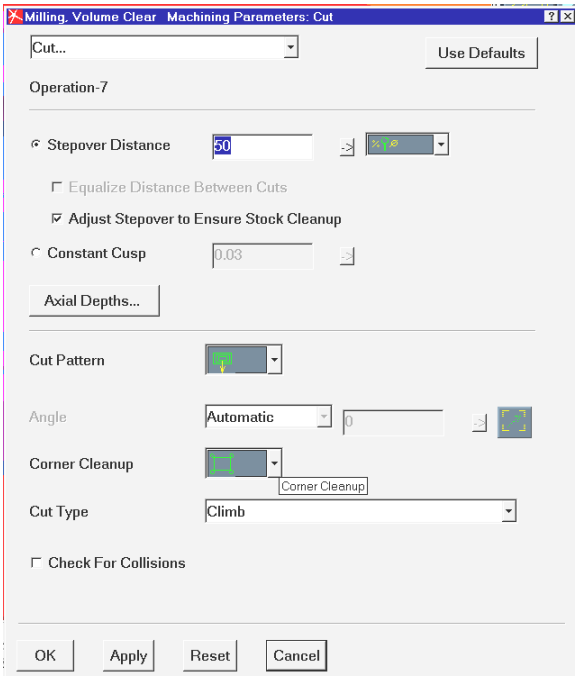
Identify the surfaces to be cut by picking  . Pick the same icon on the *Stock Specification* form that displays. Select all of the surfaces to be machined by holding down the *Shift* key while clicking on the surfaces. Once all of the surfaces are highlighted, click the MMB to signify *Done*. You can also specify surfaces to avoid or ignore, if necessary. Click *OK* when done with the form.





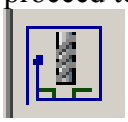
The cutting tool must be defined by picking the  . A *Cutting Tool Specification* form will display. It is suggested that you use tools from the catalog that has been stored under the **EM433-CAM** project. Click on the *Find* button and then pull down *Select Project* and select the **EM433-CAM** project.

The cutting parameters for the operation are set from the form that displays after picking the  . This form actually has several different views that can be reached from the drop-down menu button at the top. Of particular importance are the Axial Depths and the Allowances and Tolerances forms. Do not pick *OK* until you have set appropriate parameters in ach of the views of the form.

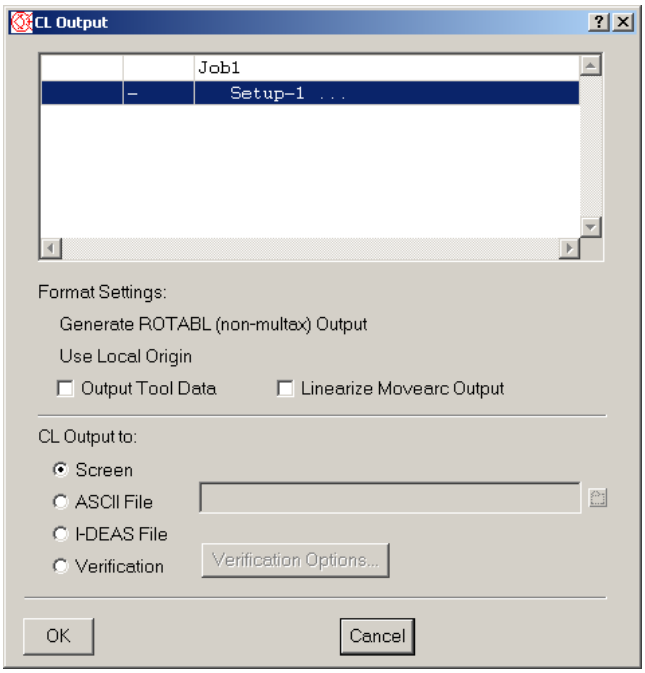


The toolpaths are actually calculated when you pick the icon to process the operation,  . A separate processor is launched and you will see a bunch of messages scroll by as it tries to decide on appropriate tool motions based on your specifications. It is not unusual to have several warning messages appear in the list region after processing the toolpath. Dismiss the Warning box and review the messages in the *List Region* to try to decide if they are critical or not. Sometimes there are suggestions on how to remedy the problem posted in the list.

You can view the tool moving through the toolpath by animating the operation with the  icon. Watch to see that the tool safely avoids your fixtures and clamps as it moves around your part. If you are satisfied with the toolpath, save your file then

proceed to specify the next operation. If not, *Modify* the operation by picking the  icon.

After all of the operations have been successfully defined, the information must be written to a CL file by picking the  icon. The CL Specification form will display.

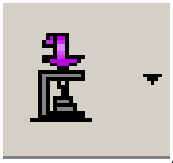


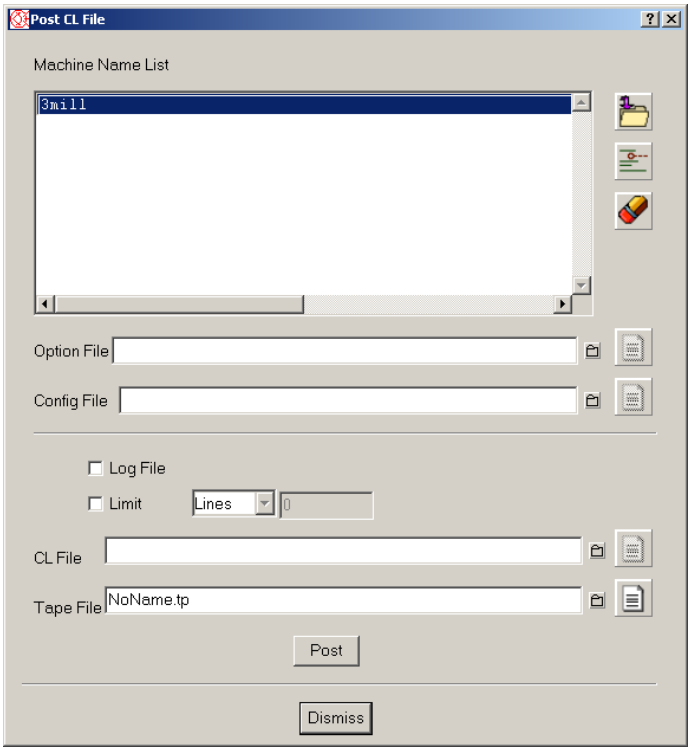
Send the output to an ASCII file and enter an appropriate file name in the box. A good idea is to name the file after the part. Give the filename a *.cl* extension.

Check the box for *Output Tool Data*.

Make sure the file is being stored in your own directory, not the Ideas temp directory!

The final step is post-processing the CL file to generate the G-code for the machine tool.

Pick the *Post CL File* icon under the *Write CL file* stack.  . A form will display with the CL file name filled out.



You must select the Option File corresponding to the machine tool controller, either S*:\ideas_nt\option\haas_mill.ppr* for the mill or S*:\ideas_nt\option\Hardinge.ppr* for the lathe and specify a name for the Tape File.

Pick the *POST* button and answer the questions that appear on the window that displays as the file is processed.

This will produce the actual tape file with the G-code that can be copied to a floppy disk and taken down to the shop to produce the part.